

## **A CUSTOMIZED AUTOCAD ENVIROMENT FOR STRUCTURAL ANALYSIS USING THREE BENDING PLATE FINITE ELEMENTS**

### **Raimundo Aprício de Menezes Jr.**

Laboratório de Ensaaios de Materiais e Estruturas, CT, Universidade Federal da Paraíba,  
Campus I, CEP 58059-900, João Pessoa, PB, Brasil.

e-mail: [jrmenezes@hotmail.com](mailto:jrmenezes@hotmail.com)

### **Moisés Salvino Menezes**

Laboratório de Ensaaios de Materiais e Estruturas, CT, Universidade Federal da Paraíba,  
Campus I, CEP 58059-900, João Pessoa, PB, Brasil.

e-mail: [riei@hotmail.com](mailto:riei@hotmail.com)

### **Faustino Sanches Junior**

Departamento de Engenharia de Estruturas, EESC, Universidade de São Paulo, CEP 13560-  
590, São Carlos, SP, Brasil.

e-mail: [faustino.Sanches.jr@uol.com.br](mailto:faustino.Sanches.jr@uol.com.br)

### **Ângelo Vieira Mendonça**

Departamento de Engenharia Civil, CT, Universidade Federal da Paraíba, Campus I, CEP  
58059-900, João Pessoa, PB, Brasil.

e-mail: [mendonca@ct.ufpb.br](mailto:mendonca@ct.ufpb.br)

**Abstract:** *This paper deals with an attempt to incorporate to standard drawing tools of Autocad package an evaluation tool related to Finite Element Analysis (FEA) of bending plate problems. Many aspects of development of the present customized environment called StructCAD are described. Firstly some remarks are addressing to pre-processing stage where analysis properties required by FEA can be added to drawing information. Then without quitting customized environment, the stiffness matrices and equivalent nodal forces can be evaluated. At last the post-processing libraries can be run in order to complete the analysis without quitting the customized environment. Numerical examples are shown in order to illustrate the feasibility of present customized package and to make comparisons among triangular bending plate elements.*

**Key words:** Structural analysis, DKT, HSM, DST, OOP.

## 1.INTRODUCTION

Engineering practice requires engineer to deal with structures containing usually high degrees of freedom so that analysis procedures can be become a cumbersome task if there is not an efficient and friendly tools to do the stages of analysis, which are defined from pre-processing to post-processing of numerical model. There are many commercial and academic packages available in engineering community and they can basically be classified into two groups: a) single structural analysis systems and b) complete structural analysis systems. In first group the available tools are restricted to determining specifically quantities such as displacements and effort fields. Majority of analysis soft wares provide Finite Element Method (FEM) calculation incorporating many mathematical models. In case of frame and slab problems, some academic [1-3] and commercial [4-7] packages can be addressed. The second group of structural analysis provides a wider spectrum of tools. In addition to calculation of quantity problem, the package is also able to check design criteria according to brazilian and/or international code for a specific material such as reinforced concrete and steel. In some packages are also available drawing tools using a CAD (Computer Aided Design) system in order to detailing stages of the structural members [8-13]. The majority of aforementioned packages handle directly with external files to import or export archives in order to provide data communication through pre-processing to processing and finally to post-processing stages. In contrast, oriented object programming strategies have been used in conjunction with drawing packages such as Autocad [16]. Santos & Devloo[17], Campagnã & Meneses[18], Maud et al.[19] describe an attempt of compatibilization between drawing environment of Autocad package and finite element analysis environment (classes in C++ for frame and bending plate problems) using ObjectArx library. However, if all stages of analysis (pre-processing, processing and post-processing) are successfully implemented in [17-19] in order to build a single customized Autocad environment is rather unclear.

In this paper is described an attempt to incorporate to standard drawing tools of Autocad package an evaluation tool related to Finite Element Analysis (FEA) of bending plate problems. Hence, other properties than element geometry such as mechanical properties, element connectivity, and boundary constraints can be added to drawing information, providing a more friendly and efficient pre-processing environment. Next without quitting customized Autocad, the stiffness matrices and equivalent nodal forces related to bending plate finite elements can be evaluated. The evaluation functions are writing in oriented object language C++ and they are compiled as dll libraries, whose can be run from a menu of customized Autocad environment. At last the analysis is completed as soon as the post-processing libraries are run. The analysis results are shown in specific dialog boxes run from customized Autocad as well. Numerical examples are shown in order to illustrate the feasibility of present customized package and to make comparisons among triangular bending plate elements well-known in literature such as Hybrid Stress Model (HSM), Discrete Kirchhoff Theory (DKT) and Discrete Shear Theory (DST).

To best Authors knowledge the present paper is the first work to put together all stages of modeling (pre-processing, processing and post-processing) into a single customized Autocad environment for bending plate analysis using three different finite elements.

## **2.PRE-PROCESSING**

The pre-processing is usually defined as a stage of the analysis where input data (such as geometry, node and element numbering, mechanical properties, boundary conditions, loading) are set to perform the calculation of the discretized problem. The original Autocad environment provides only drawing capabilities so that it is necessary to build intermediate tools in order to incorporate remaining FEM input data into drawing information. This procedure of compatibilization has usually been done using two strategies namely Autolisp

Language [20] and ObjectARX library [21]. In first approach the source code must be shown in explicit form to the users because the Autolisp is an interpreted programming language. In second approach is used the Oriented Object Programming (OOP), where ObjectARX library with more than 200 classes and 3000 objects are written in C++ programming language. It should be noted the ObjectARX library has a compiled form and therefore the source code is not necessarily to have be shown to the users.

In present work was used Autolisp language for pre-preprocessing programming in order to implement the drawing routines and to assign structural analysis data. Principal functions of pre-processing menu in StructCAD are: “Projetar Nós da Estrutura”, “Projetar Placas da estrutura”, “Define e Numera estrutura”, etc. For sake of conciseness, only Autolisp source code for function “Projetar Placas da estrutura” is shown in Table 1. For information about remaining Autolisp functions in StructCAD, the reader is addressed to Menezes Jr [24].

Table1- Autolisp code for “Projetar Placas da estrutura” function

1. ;;ProjetaPlacas	30. (action_tile "N_elemY" "(setq \$N_elemY \$value)")
2. ;;(ProjetaPlacas)	31.
3. (defun ProjetaPlacas ()	32. (setq OK (start_dialog))
4.	33. (unload_dialog DH)
5. (setq dh (load_dialog	34.
"C:\Amate\Suporte\m_placas.dcl"))	35. (if (= OK 1)
6. (if (not (new_dialog "ProjPlacas" dh))	36. (progn
7. (exit)	37.
8. )	38. (princ "\nDefina a placa através da sua diagonal.")
9.	39. (setq Vrtc1 (getpoint "\nPosicione o Vertice Inicial:
10. (setq OK 0)	"))
11.	40. (initget 32)
12. (if (not \$N_Placa)	41. (setq Vrtc2 (getcorner Vrtc1 "\nPosicione o Vertice
13. (setq \$N_Placa "1")	Oposto: "))
14. )	42. (princ "\n")
15.	43.
16. (if (not \$N_elemX)	44. (setq NelemX (atoi \$N_elemX)
17. (setq \$N_elemX "1")	a. NelemY (atoi \$N_elemY)
18. )	45. )
19.	46.
20. (if (not \$N_elemY)	47. (DefinePlaca)
21. (setq \$N_elemY "1")	48.
22. )	49. )
23.	50. )
24. (set_tile "N_Placa" \$N_Placa)	51.
25. (set_tile "N_elemX" \$N_elemX)	52. (princ)
26. (set_tile "N_elemY" \$N_elemY)	53. )
27.	54. ;;end
28. (action_tile "N_Placa" "(setq \$N_Placa \$value)")	
29. (action_tile "N_elemX" "(setq \$N_elemX \$value)")	

When the user selects the menu option “projetar placas”, the dialog Box “Projetar elemento finito de placa” is shown in the customized environment, so that this corresponds to execute the line 5 instructions in source code shown in Table 1. After setting dialog box values, the user must press the “Ok” button to apply the input data (see Figure 2b). This procedure executes source code line 47 in Table 1, resulting in discretized form of plate (see Figure 3).

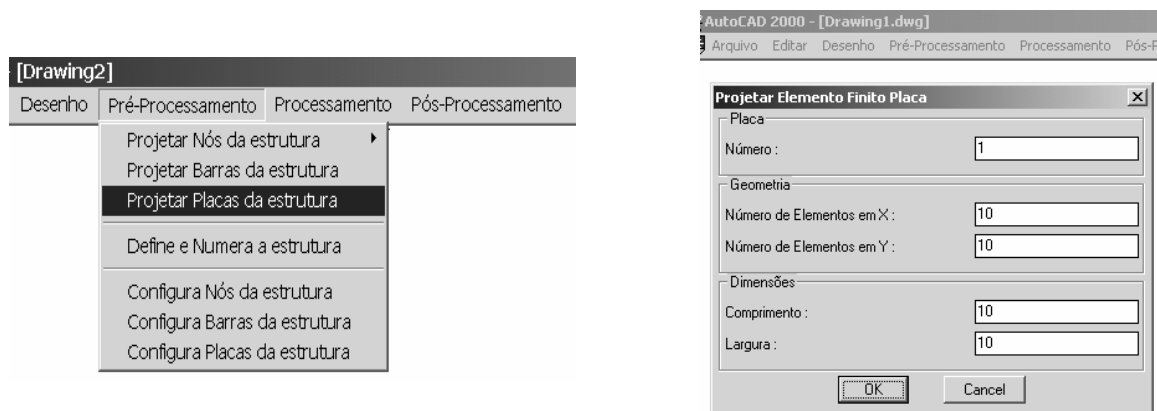


Figure 2 – (a) Projetar Placas da estrutura; b) Projetar Elemento Finito Placa

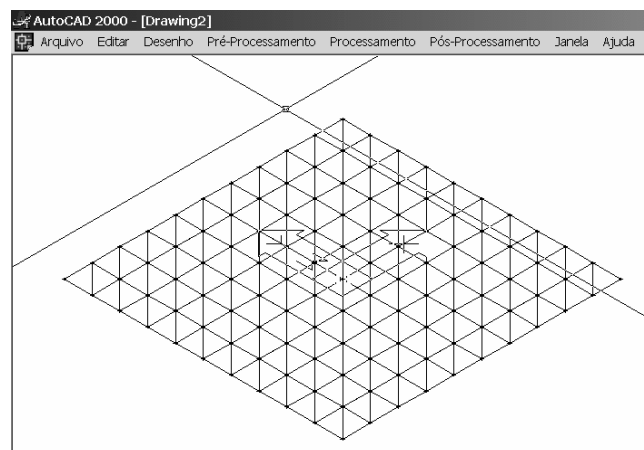


Figure 3 – Discretized Model

Another step in StructCAD pre-processing is related to the definition of structural element type and element connectivity. Theses procedures can be done by selecting the StructCAD option menu “Pré-processamento->Define e Numera a Estrutura” (see Figura 4-a) and by selecting the dialog Box “Tipo” (see Figure 4-b).

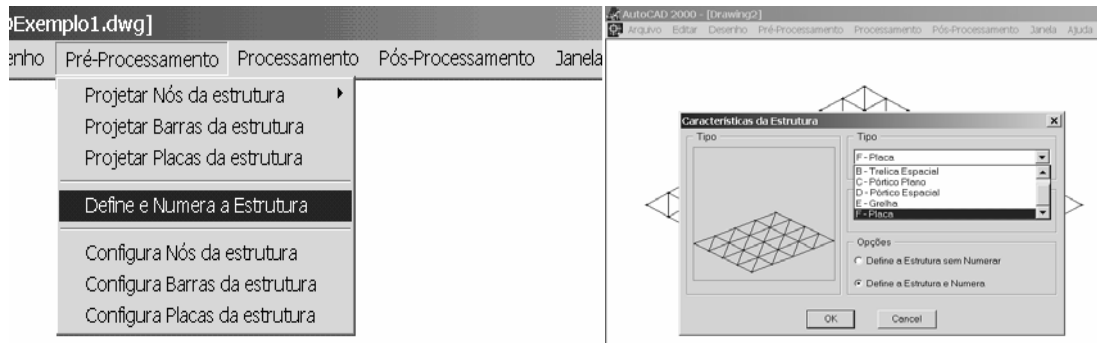


Figura 4 – (a) “Define estrutura” function; b) “Tipo” function.

### 3. PROCESSING

The processing is usually defined as a stage of the analysis in which main calculations (such as elemental stiffness matrix evaluation, structural stiffness matrix assembling, nodal equivalent force vector evaluation, algebraic system solution, etc) are done. In present paper, the bending plate problem is analysed by Finite Element Method (FEM) using three flat triangular elements namely DKT (discrete Kirchhoff Theory), HSM (hybrid stress model) and DST (Discrete Shear Theory). The first two elements were developed to deal with thin bending plate problems and their formulations have been thoroughly discussed elsewhere Batoz et al.[22]. The DST element takes into account the deformation by shear force and it was developed by Batoz & Lardeur[23] based on Refined Theory of Plate. The final nodal variables used in the formulation of three elements are represented by two rotations and one translation in each corner of the triangle as shown in Fig. 5

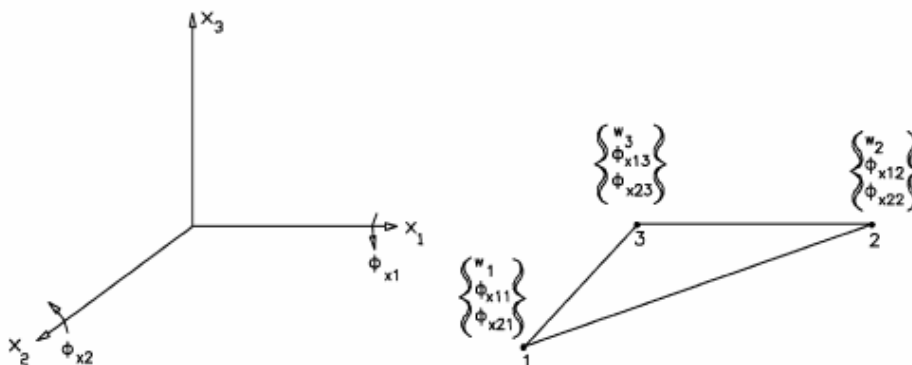


Figure 5- Nodal quantities of HSM, DKT and DST elements

The bending plate analysis is done using the CaeFep++ library, in which have functions and classes implemented in oriented object language C++ and they are compiled as ObjectARX program, whose it can run from a menu of the StructCAD. For sake of conciness, only principal classes and functions of the CaeFep++ library are described next.

The class diagram of CaeFep++ (Computer Aided Engineer for Finite Element Plate) library is shown in Figure 6. It should be noted that the CaeFep++ has functions and classes associated with three plate bending finite elements namely DKT, HSM and DST.

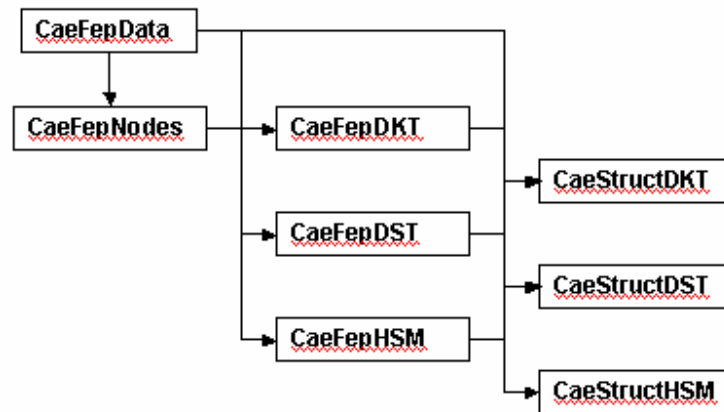


Figure 6- CaeFep++ class diagram

For sake of conciseness, only fields and methods related to the classes CaeFepDST and CaeStructDST are respectively shown in Table 2 and Table 3.

Table 2- The CaeFepDST class

Class	::CaeFepDST
Object	Void CalculaAreaDST(int NumDst)
Object	Void GeraGeometriaDST (int NumDst)
Object	Void GeraIncidenciaDSTs ( )
Object	Void GeraMatrizRigidezLocalDST (int NumDst)

Table 3- The CaeStructDST class

Class	::CaeStructDST
Object	Void AplicaCondicaoDeContorno ( )
Object	Void EliminadorDeGauss ( )
Object	Void EsforçosNoDST ( )
Object	Void GerarMatrizDeRigidezDaEstrutura ( )
Object	Void GerarVetorForçasNodais ( )

Object	Void ResolveEstrutura ( )
--------	---------------------------

In general, external functions can only be accessed from Autocad environment if these functions are compiled as ObjectARX format. In present work “solver” is a function defined as main function that uses methods and objects from ObjectARX library and CaeFep++. The “solver” function works in two steps: in the first the objects and functions of ObjectARX library collect the input data assigned in pre-processing stage, and store them in vectors. In the second step, these vectors are processed by objects and methods from CaeFep++ to perform the structural calculations. It should be noted that CaeFep++ must be compiled in conjunction with ObjectARX library in order to create the function “solver”, who after compilation process, it is stored in a file called “Aeplaca.arx”. Then, for the structural tool to be run from the StructCAD, the file “Aeplaca.arx” must to be loaded to it. It should be noted that at moment of loading “Aeplaca.arx” file, the method “entrypoint” of the objectARX library is called in order to become accessible the function “solver” to Autocad environment. The source code of “entrypoint” function can be found in ref.[21].

Form a point of view of user, all processing steps are done when he selects the menu option “processamento -> Calcular Estrutura” (see figure 7-a). It should be noted that processing stage is done without quitting customized Autocad environment StuctCAD. In addition, a warning flag is displayed to the user as soon as processing is done (see figure 7-b).

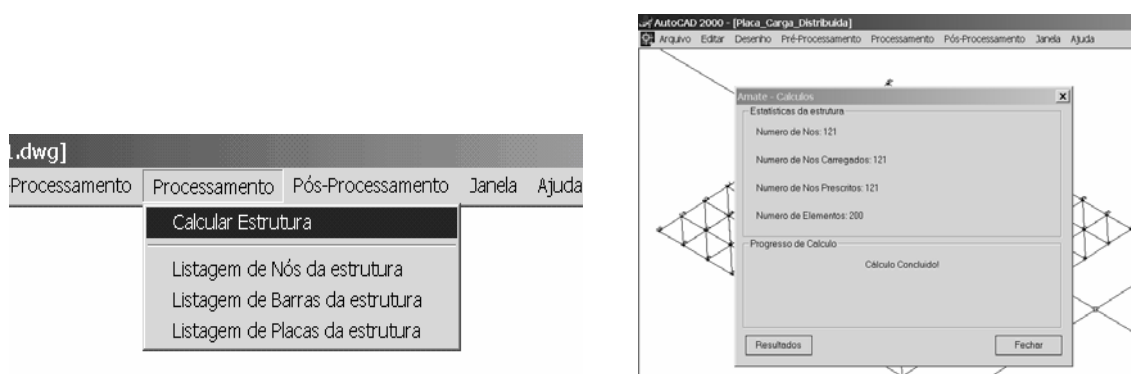


Figure 7 – (a) “Calcular Estrutura” function

b) Warning flag



## 4. POST PROCESSING

The Post-processing is usually defined as a stage of the analysis in which the results are shown. In StructCAD the results can be accessed by selecting the menu option “Pós-Processamento -> Esforços e Deslocamentos da Estrutura” (see Figure 9-a) and the values are shown in dialog box (see Figure 9-b)

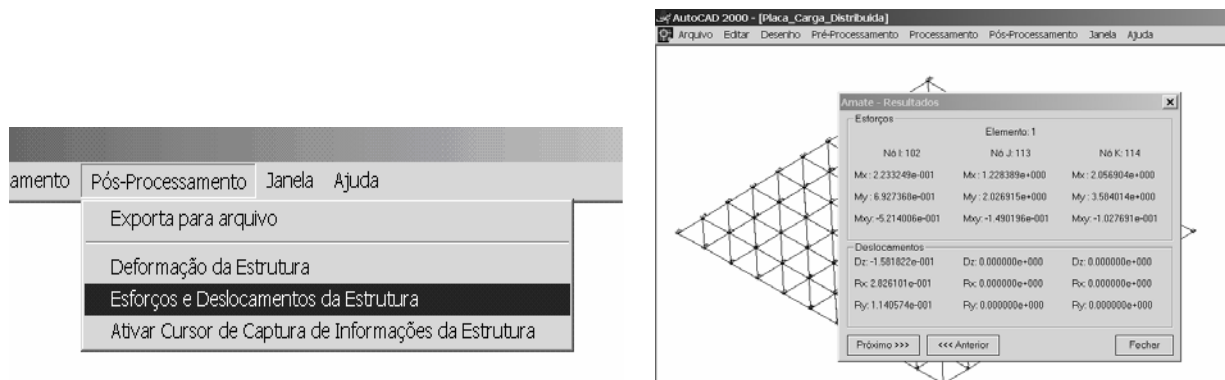


Figure 9 – a) “Esforços e Deslocamentos da Estrutura” function; .b) Dialog Box of results

It should be noted that buttons “>>> Próximo” and “<<< Anterior” provide the result selection for a next or previous element to be shown in dialog Box (see Figure 9-b).

The StrucCAD provides a graphical output for displacements by selection of menu option “Pós-Processamento-> Deformação da Estrutura” (see figure 10-a) and the deformed structure form is shown in Figure 10-b

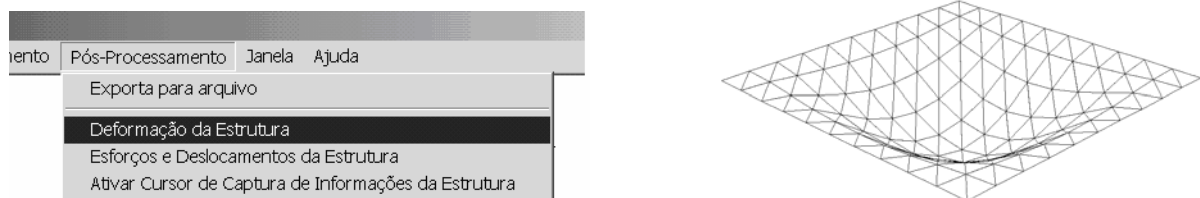


Figure 10 – a) “Deformação da estrutura” function; b) Deformed shape.

## 5. NUMERICAL ANALYSIS

In this section a simply supported square plate under concentrated loading is analysed. The plate has unitary side and Young modulus  $E = 1092 \text{ MPa}$  and Poisson ratio  $\nu = 0.3$  for mechanical properties and a concentrated load  $P = 1 \text{ kN}$  is applied at the plate centre. The plate discretization and node numbering are shown in Fig. 11.

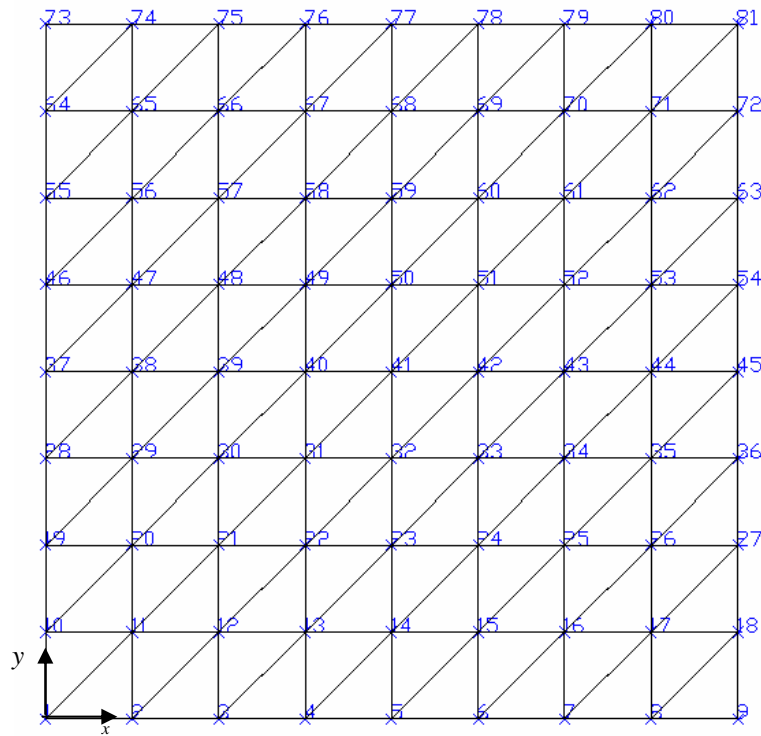


Figure 11- Plate discretization

For concentrated load, the transverse displacements  $w$  and bending moments  $m_x$  (along direction  $x$  and central line of plate for different span/thickness ratios) are shown in Tables 4 and 5. It should be noted that in Classical Plate Theory the bending moments are insensitive to flexural rigidity  $D = Et^3 / [12(1 - \nu^2)]$  where  $E$ ,  $t$ ,  $\nu$  are respectively Young modulus, thickness and Poisson ratio.

Table 4- Transverse displacements along direction  $x$  (central line of plate)

		w (m)		
L/t	Node	HSM	DKT	DST
10	37	.0000E+00	.0000E+00	.0000E+00
	38	.3658E-04	.3681E-04	.3842E-04
	39	.7100E-04	.7169E-04	.7522E-04
	40	.1001E-03	.1013E-03	.1081E-03
	41	.1139E-03	.1164E-03	.1302E-03
30	37	.0000E+00	.0000E+00	.0000E+00
	38	.9879E-03	.9941E-03	.9989E-03
	39	.1917E-02	.1936E-02	.1946E-02
	40	.2702E-02	.2736E-02	.2755E-02
	41	.3077E-02	.3145E-02	.3184E-02
50	37	.0000E+00	.0000E+00	.0000E+00
	38	.4572E-02	.4601E-02	.4609E-02
	39	.8874E-02	.8961E-02	.8977E-02
	40	.1251E-01	.1266E-01	.1269E-01
	41	.1424E-01	.1455E-01	.1462E-01
70	37	.0000E+00	.0000E+00	.0000E+00
	38	.1254E-01	.1262E-01	.1264E-01
	39	.2435E-01	.2459E-01	.2461E-01
	40	.3432E-01	.3474E-01	.3478E-01
	41	.3908E-01	.3994E-01	.4002E-01

Table 4- Bending moments along direction x (central line of plate)

		$m_x$ (kNm)		
L/t	Node	HSM	DKT	DST
10	37	-.3263E-03	.6069E-02	-.2173E-02
	38	.3135E-01	.2814E-01	.3231E-01
	39	.5982E-01	.5904E-01	.6300E-01
	40	.1282E+00	.1228E+00	.1307E+00
	41	.3049E+00	.3266E+00	.3202E+00

	Node	DST (L/t = 30)	DST (L/t = 50)	DST (L/t = 70)
	37	.2804E-02	.4590E-02	.5256E-02
	38	.2980E-01	.2890E-01	.2856E-01
	39	.6042E-01	.5964E-01	.5937E-01
	40	.1257E+00	.1241E+00	.1235E+00
	41	.3241E+00	.3255E+00	.3260E+00

The results suggest that influence of shear deformation affects more severely the transverse displacements than bending moments. When the  $h/t=10$ , differences greater than 10% at central node for displacements are obtained by DST and the remaining two elements. In contrast, bending moments for the same ratio have differences about 2%.

## CONCLUSIONS

In this paper a customized Autocad environment called StructCAD was presented. The main attractive feature of this analysis system is that all stages (from pre-processing to post-

processing) can be made using integrated environment without quitting the customized Autocad.

## ACKNOWLEDGEMENTS

The Scientific Initiation Scholarships (PIBIC) sponsored by Brazilian National Council of Research (CNPq) are gratefully acknowledged by first two authors.

## REFERENCES

- [1]FTOOL. [www.tecgraf.puc-rio.br/ftool](http://www.tecgraf.puc-rio.br/ftool) . Visited in 4/20/2006.
- [2]ACADFRAME. [www.set.eesc.usp.br/acadframe](http://www.set.eesc.usp.br/acadframe). Visited in 4/20/2006.
- [3] M.A. Ramalho. Sistema para análise de estruturas considerando interação com o meio elástico. São Carlos. Tese (Doutorado) - Escola de Engenharia de São Carlos, Universidade de São Paulo, 1990.
- [4]ANSYS. [www.ansys.com](http://www.ansys.com). Visited in 4/20/2006
- [5]ABAQUS. [www.abaqus.com](http://www.abaqus.com). Visited in 4/20/2006
- [6]LUSAS. [www.lusas.com](http://www.lusas.com). Visited in 4/20/2006.
- [7] CYPECAD - Manual do usuário. Multiplus S/C Ltda. São Paulo/SP, 1997.
- [8]AltoQi Eberick versão 2002. AltoQI Tecnologia em Informática Ltda. Florianópolis/SC, 2002. Conjunto de programas. 1 CD-ROM.
- [9]. Sistrut 4.0 - Manual do usuário. Sistrut Software e Tecnologia S/C Ltda. 1992.
- [10]TQS - Manual do usuário. TQS Informática Ltda. São Paulo/SP, 1997.
- [11]Strad-Manual do usuário. 2003, Conjunto de Programas. 1CD-ROM
- [12] SAP 2000 – Integrated finite elements analysis and design of Structures, CSI – Computer and Structures, Inc. Berkeley/CA, 1997. Conjunto de programas. 1 CD-ROM.

- [13] R. H. Peixoto. Um sistema gráfico-interativo para análise e dimensionamento de pórticos planos de aço. Dissertação Mestrado. PUC-Rio, 2001.
- [15] A.A. Bortolin. Sobre a racionalização e a automação do cálculo de edifícios de concreto armado. São Carlos. Dissertação (Mestrado) – EESC –USP, 1991.
- [16] AUTODESK INC. *AutoCAD 2000i - User's Guide*, 1999.
- [17] E. S. R. Santos, P. R. B. Devloo. Desenvolvimento de classes viga e pilar para autocad utilizando linguagem C++ e objectarx em “ IX Congresso Interno de Cientifica da Unicamp”, 27 e 28 de setembro, 2001.
- [18] F. H. C. Mauad, E. S. R. Santos, Phillipe Remy Bernard Devloo. Integração de Métodos Numéricos no Autocad, utilizando o Objectarx e o Ambiente de Elementos Finitos – Pz. In: Jornadas Sul-Americanas De Engenharia Estrutural, 30, 27 A 31 De Maio De 2002 – Universidade de Brasília, – Brasil
- [19] CAMPAGNÃ, B. C. D. ; MENEZES, F. A. . Customização do auto cad utilizando-se a linguagem c++ e o objectarx - Dimensionamento de lajes de concreto. In: IX Congresso interno de iniciação científica da Unicamp, 2001
- [20] Kramer, W. e Kramer, D. Programando em AutoLISP. MAKRON Books do Brasil Editora Ltda, São Paulo - SP, 1995.
- [21] Charles McAley. *Programming AutoCAD 2000 Using ObjectARX*. Autodesk Press, 2000.
- [22] Batoz JL, Bathe KJ, Ho LW. A study of three-node triangular plate bending elements. Int J Numer Meth Engng 1980;15:1771–812.
- [23] Batoz JL, Lardeur P. A discrete shear triangular nine d.o.f. element for the analysis of thick to very thin plates. Int J Numer Meth Engng 1989;28:533–560.

[24] Menezes Junior, R. A. “Análise da interação dos efeitos placa-pórtico em problemas de edifícios utilizando-se programação orientada ao objeto em C++. PIBIC 2005-2006, Universidade Federal da Paraíba, João Pessoa, PB.